COMPUTATIONAL MODELING OF MARINE PROPULSORS

Charles L. Merkle and Jinzhang Feng
Department of Mechanical Engineering
The Pennsylvania State University
University Park, PA 16802

FINAL REPORT

Mr. Gerald Smith
Regional Director
Office of Naval Research
Federal Building
Second Floor, Room 208
536 South Clark Street
Chicago, IL 60605-1588

Grant/Contract No. N00014-91-J-4058

March, 1997

PENNSTATE

(814) 863-6272
FAX: (814) 865-3389

Propulsion Engineering Research Center

The Pennsylvania State University
106 Research Building East — Bigler Road
University Park, PA 16802-2320

March 11, 1997

Mr. Gerald Smith
Regional Director
Office of Naval Research
Federal Building
Second Floor, Room 208
536 South Clark Street
Chicago, IL 60605-1588

Re: Grant/Contract No. N00014-91-J-4058
"Computational Modeling of Marine Propulsors"

Dear Mr. Smith:

Enclosed is a copy of our Final Report on the above-referenced grant as per contract guidelines. Please accept my apology for the delay in processing this report.

If you have any questions, please do not hesitate to contact me at (814)863-1501 or by fax at (814)865-3389.

Very truly yours,

Charles L. Merkle

Charles L. Merkle
Professor of Mechanical Engineering

CLM:ca
Enclosures

cc: Dr. Edwin P. Rood (3 copies)
Naval Research Laboratory (1 copy)
Defense Technical Information Center (2 copies)
F. Ramos, ONR (cover letter only)
PSU COE Research Office (cover letter only)

Computational Modeling of Marine Propulsors

The goals of this research effort are to model the flow through marine propulsors and turbomachinery systems with three distinct subsets of the equations of motion, namely the velocity potential equations; the Euler equations; and the Navier-Stokes equations. Each of the three sets of equations has a specific role in the description, the understanding and the control of the complex flow field of marine propulsion applications.

This research of five years has accomplished its goals as proposed. First, within the velocity potential framework, a time-accurate algorithm has been developed to model rotor-stator blade interaction, including wake vortex generation and tracking, blade-vortex interaction, and finally multi-row moving blade interaction. Further, this model is also used to study blade passage noise generation for marine turbomachinery. The panel-based velocity potential approach developed remains an economical alternative to Navier-Stokes solutions, which can be effectively used for preliminary numerical study of marine propulsion flow fields and for initial design of marine propulsors.

Second, through this research effort, a multi-block, multi-grid Navier-Stokes solver has been developed with various turbulence model options. This Navier-Stokes solver has been used, in this research effort, to study rotor-stator interactions, blade tip clearance effects in marine turbomachinery applications and related problems. This Navier-Stokes solver has also been transitioned to the Navy and is now in use by our colleagues at DTMB to study flow field of various other propulsion geometries.

Third, based on the potential panel method and the Navier-Stokes solver, a new design approach has been developed for low-speed, high pressure rotor-stator systems. A series of such rotor stator fans has been successfully designed using this approach for on-board ventilation applications of Navy ships, resulting in a 15 db reduction of noise level and enhanced performance for these fans.

In addition to Navier-Stokes solutions, Euler solutions can also be obtained by turning off the viscous terms in the Navier-Stokes solver. Euler solutions require reduced numerical efforts, but improvements in computer capabilities are beginning to make the Euler solution option less valuable today than as at the start of this research effort.

A brief description of specific technical achievements within this research effort follows.

Part One, Computational Modeling with Velocity Potential Equations

The velocity potential equations offer a relatively economical solutions for inviscid irrotational flows by representing the flow field with the superposition of panel singularities on the boundary surfaces. By also placing such boundaries on

wake (vortex) surfaces, in addition to the body surfaces, we are also able to obtain useful solutions for a number of limited vortical flows, such as trailing edge induced vortical flows, blade-vortex interactions, etc. With vortex tracking and numerical integration techniques, a time-accurate panel code has been developed to solve flow fields containing multiple moving bodies, such as the case of multi-row moving cascades.

Mathematical Description and Numerical Procedures

On the basis of Green's Second Identity, a combined singularity distribution of uniform source and piecewise linear doublet are placed on appropriate boundary surfaces. A pressure type Kutta condition is enforced at the blade trailing edge. Time accurate vortex tracking and integration are used to model the convection of vortex in the flow field, such as the convection of blade wake vortex. Another important aspect of the numerical scheme is to establish a closed expression of the Green function for cascade configurations. We have found that using an algebraic superposition of a limited number of single bodies to approximate the cascade geometry can lead to erroneous results. For brevity, mathematical descriptions and numerical procedures are not presented here in this report. Detailed descriptions can be found in the following published papers [1-5].

Application of Velocity Potential Approach to Marine Propulsion Problems

Panel based velocity potential approach has been used widely to predict steady state pressure distributions over body surfaces. The primary focus of this research is to extend this approach to study unsteady rotor-stator interactions and their acoustic noises. This study has three components. The first is unsteady rotor dynamics, which includes moving blades with vortex generation [1, 2]. The second is blade vortex interaction that is capable of capturing blade wake-cutting [3,4]. And finally, the time-dependent rotor-stator interactions, including wake vortex generations from the blade trailing edges, wake vortex convection/rolling-up from upstream to downstream, blade wake-cutting, and blade-blade interactions [5]. Some of the representative numerical computations include the UTRC rotor-stator configurations of various ratios of rotor/stator blade numbers (one-to-one, two-to two, four-to-three, and six-to-five), the MIT flapping hydrofoil experiments, the HIREP rotor-stator configurations, etc.

In conjunction with other research projects, this velocity potential approach is also used to model cavity flow problems [6]. In this approach, the cavity surface is considered as a unknown streamline of a constant pressure. The constant pressure condition together with the streamline conditions are used iteratively to obtain the cavity location as well as the whole flow field.

Part Two, Computational Modeling with the Navier-Stokes Equations

The Navier-Stokes equations enable the full effects of viscosity, turbulence, separation, recirculation and other flow details to be included in the computational modeling. Some important aspects of the development of the Navier-Stokes solver include preconditioning for incompressible flow systems [7,8]; multi-block representation [9]; overset grids for complex geometries; multi-grid acceleration techniques; and various turbulence model options, including an algebraic model (Baldwin-Lomax), a one-equation model (Baldwin-Barth), a k-e model and a k-e-v2 model (Durbin). Parallel computing is also implemented over a cluster of distributed workstations [10].

The developed Navier-Stokes solver is based on an artificial compressibility formulation with multi-stage Runge-Kutta temporal integration and second-order central differencing in space. Fourth order artificial dissipation is used to prevent odd-even splitting. Again, detailed descriptions of these numerical procedures can be found in the referenced papers and are not repeated here.

Application of Navier-Stokes Solver to Marine Propulsion Problems

The Navier-Stokes solver is used to study the flow details of marine turbomachinery and rotor-stator interactions. These details include blade surface pressure distributions, wake distribution, flow separation, various vortex structures and gap flow in the blade tip region. A more detailed description of these studies is presented in the following representative examples.

Rotor-Stator Interaction [11]

The Navier-Stokes solver is used to study flow in the HIREP stator-rotor configuration. Detailed measurements of the flow field at several stations and the pressure distributions were carried out at ARL in Penn State. These measurements provide a comparison base for our numerical computation. Without going into the details of the numerical results at length, our computation can be summarized as follows,

a. The pressure distribution from the numerical solution compares very well with the experimental measurements.

b. The blade wake distribution compares well with the experimental data over most part of the blade radius. Near the stator/rotor hub, however, the boundary thickness is over-estimated compared with experimental measurements. Near the rotor blade tip, a secondary wake caused by the tip gap is not completely captured. These errors may be due to insufficient grid resolutions in both regions as well as the lack of gap flow modeling.

c. A radial vortex structure is present on the suction side of the rotor blade nearing the blade trailing edge. This radial vortex is also observed in the water tunnel experiment. Near the blade tip, this radial vortex merges with the tip gap vortex to form a strong blade tip vortex that convects into the blade wake. This vortex is identified to have significant impact on the hydrodynamic performance as well as the cavitation performance of the rotor.

d. A relatively large region of separation exists on the suction side of the rotor blade near the trailing edge.

The Effect of Tip Clearance [9]

Tip gap flows have a significant effect on turbomachinery performance. The details of gap flows consist of the gap separation, the passage vortex, the gap vortex and the trailing radial vortex.

Two tip gap flows have been studied with an embedded grid structure. One is a stationary three-dimensional cascade. And the other is the HIREP rotor. Existing experimental measurements are available for both cases.

In the first case, the three dimensional cascade is heavily loaded (the flow turning angle is 45 degrees between the exit and the inlet). The blade tip gap height is three percent of the blade span. The gap leakage flow is quite strong because of the heavy loading on the blade. Numerical solutions have clearly captured the flow separation inside the gap, the passage vortex and the tip gap vortex. In addition, a pressure reloading in the mid-chord region close to the blade tip is also present in the numerical computation as in the experiments.

In the rotor blade case, three different gap spaces have been used to evaluate the tip gap effects on the flow field. Since the blade loading is not as high as in the cascade case, some of the flow phenomena associated with the tip gap that appeared in the cascade case are not observed in the rotor case. With the increase of the tip gap height, enhanced mixing between the suction side and the pressure side is facilitated, thereby resulting in additional load reduction on the blade.

Cavitation Modeling Using Navier-Stokes Solvers [12,13]

In conjunction with other research projects, the Navier-Stokes solver has also been used to model cavitating flow over blade surfaces. The cavity model is analogous to the stream line model used in the previously discussed panel approach. The boundary conditions employed at the cavity surface are shear-stress free conditions augmented by a constant pressure condition and a zero-normal velocity condition. Dynamical re-gridding is used to adapt the computational domain to the latest cavity surface at the end of each iteration. Numerical comparisons are in good agreement with available experimental measurements for a number of partial cavity problems of flow past an hydrofoils.

Part Three, Rotor-Stator Design Using CFD [14, 15]

The above discussed panel-based velocity potential approach and the Navier-Stokes solver have been combined to re-design a series of high pressure, low speed rotor-stator fans. The original designs of these fans could not deliver stable performance over a specified working range and generated too much noise. The panel code was used to quickly design the blade geometry by aligning the rotor and stator blade leading edges with the flow field as well as enforcing the blade loading requirements. The Navier-Stokes solver was used as a final check to examine details of the flow field for the designed geometry. One of the important details is to ensure that flow separations do not occur anywhere on the blade surfaces. Other aspects of the design were to estimate the effects of simplified design configuration on the performance of the rotor-stator fan. One example of such computations was a study to estimate the performance degradation incurred by replacing the hydrofoil shaped stator blade with a rounded flat plate which is more economical to manufacture. Full scale experiments have demonstrated that this design effort has produced a series of fans that reduce the fan operational noise by 15 db while enhancing their performance.

Summary

At the start of this research effort, extensive numerical computations with Navier-Stokes equations for marine propulsors were not routinely conducted anywhere. Now with the advent of much improved computer resources, these computations are carried out regularly to study the complex flow fields, to improve existing designs, and to guide new designs. Looking ahead, computational modeling of marine propulsors has turned a new page. More efficient, more reliable and more robust N- S solvers with more physical submodels are needed for a CFD approach to be fully integrated into routine design procedures. Time-accurate Navier-Stokes computations are needed to predict acoustic noises in a marine propulsion system. Furthermore more advanced turbulence modeling, such as LES, needs to be included in these computations. Nevertheless, computational modeling of marine propulsors has already started to bear fruits for a better and more economical propulsor design.

REFERENCES

1. Feng, J. and Lee, Y.T., "Flow Calculation for Linear Cascades," Unstead Flow Forum, ASME 1990 Winter Annual Meeting, Dallas, TX, December, 1990.

2. Feng, J., Merkle, C.L., Lee, Y.T., and Bein, T.-W., "Unsteady Rotor Dynamics in Cascade," Journal of Turbomachinery, Vol. 115, No. 1, pp. 85-93, January 1993.

3. Feng, J. and Merkle, C.L., "An Interaction Noise Between Vortex and Airfoil," Paper No. AIAA-93-0600, 31st Aerospace Sciences Meeting & Exhibit, Reno, NV, January 11-14, 1993.

4. Lee, Y.T., Feng, J., and Merkle, C.L., "Prediction of Vortex and Linear Cascade Interaction Noise," ASME Paper 93-GT-314, International Gas Turbine and Aeroengine Congress and Exposition, Cincinnati, OH, May 24-27, 1993.

5. Lee, Y.-T., Feng, J., and Merkle, C.L., "Time-Dependent Potential Flow Analysis of Rotor-Stator Systems," 6th International Conference on Numerical Ship Hydrodynamics, Iowa City, IA, August 2-5, 1993.

6. Feng, J., Lee, Y.T., and Merkle, C.L., "Cavity Flow Analysis Using Pressure Coupled Panel Method," International Association for Boundary Element Methods Annual Meeting and Symposium, Boulder, CO, August 2-5, 1992.

7. Merkle, C.L. and Feng, J., "Evaluation of Preconditioning Methods for Time-Marching Systems," AIAA Paper No. 90-0016, AIAA 28th Aerospace Sciences Meeting, Reno, NV, January 8-11, 1990.

8. Merkle, C. and Feng, J., "A Unified Time-Marching Procedure for Compressible and Incompressible Flows," Journal of Hydrodynamics, Ser. B., Vol. &, No. 4, pp. 13-21, 1995.

9. Lee, Y.-T., J. Feng, and C.L. Merkle, "Effects of Tip Clearance Flows," 21st Symposium on Naval Hydrodynamics, Trondheim, Norway, June 24-28, 1996.

10. Deshpande, M., Feng, J., Merkle, C.L., and Deshpande, A., "Application of a Distributed Network in Computational Fluid Dynamic Simulations," Journal of Supercomputer Applications, Vol. 8, No. 1, pp. 64-67, Spring 1994.

11. Lee, Y-T., Feng, J., Merkle, C.L., Hah, C., and Loellbach, J., "Steady and Unsteady Stator-Rotor Interactions," Symposium on Turbomachinery Noise, 1995 International Mechanical Engineering Congress & Exposition, San Francisco, CA, November 12-17, 1995.

12. Deshpande, M., Feng, J., and Merkle, C.L., "Cavity Flow Predictions Based on the Euler Equations," Journal of Fluids Engineering, Vol. 116, No. 1, pp. 36-44, March, 1994.

13. Deshpande, M., Feng, J., and Merkle, C.L., "Numerical Modeling of Thermodynamical Effects on Cavitation," accepted for publication in the June issue of the Journal of Fluids Engineering.

14. Lee, Y.T., Feng, J., Slipper, M.E., and Merkle, C.L., "Design of an Incompressible High-Pressure Vaneaxial Fan Using CFD," 37th ASME International Gas Turbine and Aeroengine Congress and Exposition, Köln Messe, Cologne, Germany, June 1-4, 1992.

15. Lee, Y.-T., J. Feng, and C.L. Merkle, "Redesign of High-Pressure Ventilation Fans," ASME-95-CTP-73, ASME Cogen-Turbo Power Conference, Vienna, Austria, August 23-25, 1995.